



## John T. Anderson Engineering Note

**Date:** November 19,1998

**Rev Date:** November 19,1998

**Project:** General

**Doc. No:** a981119b

**Subject:** Linking Orcad Schematic Designs to Layout and Procurement

### ***Introduction***

This note is a continuation of my previous engineering note #a981007a, [How to design with Orcad](#). In this note I intend to show how forethought in the schematic entry process minimizes the effort in bringing the board forward into layout, and how to integrate Orcad with the Parts & Vendors program to minimize relookups of procurement data.

### ***Using the Bill of Materials to Advantage***

The Orcad Bill of Materials program provides a very convenient way to analyze the schematic. Once the data entry is complete and the schematic passes the Design Rules Check, the Bill of Materials is used to extract the component data and prepare for both layout and procurement. To utilize the Bill of Materials to advantage requires understanding of Orcad properties and how they're processed to build the BOM.

Every component placed on the schematic has a set of Orcad pre-defined properties and any number of user-defined properties. To see the properties of a component, select it in the schematic and hit Ctrl-E. The following dialog box shows a typical example:

**Edit Part**

Part Value: 1

Part Reference: C32

Primitive: ☒ Default ☐ Yes ☐ No

Graphic: ☒ Normal ☐ Convert

Packaging: Parts per Pkg: 1 Part: [ ]

PCB Footprint: 1206

☐ Power Pins Visible

H:\TRIGGERELECTRONICS\CAE\CFT\_FE\_AXIAL\ORCAD\MCM\_TEST\CFT.OLB

OK Cancel User Properties... Attach Implementation... Help

Various Orcad pre-defined properties are listed here:

- The **Value**
- The **Reference**
- The **PCB Footprint**
- And less useful ones like the **Primitive**.

The Value, Reference and PCB Footprint are extremely useful for building a Bill of Materials. The combination of **Value** and **PCB Footprint** defines a uniquely procured item, whereas the combination of all three defines a unique instance of a uniquely procured item. The count of how many **References** have a given combination of **Value** and **PCB Footprint** is how many of the part you have to buy per board. The **PCB Footprint** has to be considered in the procurement decision because any design may have different parts with the same **Value**; for example, a board might have a .01 uF, 16 volt, 1206 SMT capacitor as a decoupling part at the same time it also has a .01uF, 5000 volt, radial lead high voltage isolator capacitor. In both cases the value is .01uF, but the footprint is very different.

In very rare cases extended information like tolerance or working voltage may be necessary to uniquely identify a part. To pass this information forward, the user defines a User Property by hitting the User Properties button, defining a property name and entering a value for it. If you ever define a User Property, make sure you remember what it is – give it a descriptive name.

## Creating the Bill of Materials

Once all the data is in the schematic, the BOM is created by using the **Tools...Bill of Materials** command from the main Orcad screen. First select the design file in the project view, using physical mode if the design is a complex hierarchy. Insure that the design has been fully annotated and design rule checked prior to creating the BOM. When the command is invoked, a dialog box like this appears:

### **The precise format of the Combined Property String is critically important.**

The schematic properties to be exported are enclosed in curly braces, separated by the '\t' symbol (which should be familiar to C programmers as the control string to create a tab character). **Note, however, that the Value and PCB Footprint are separated not by the tab character, but by a single forward slash.** This isn't the default that Orcad gives you – make certain it is entered correctly.

Select a convenient location for the output file. I recommend keeping it with all other CAE files for your project.

The format selected will create an output file that looks like this:

CFT Axial Project; MCM Test Board Revised: Thursday, November 19, 1998  
TBD Revision: A

Bill Of Materials November 19,1998 16:19:37 Page1

Item	Quantity	Reference	Part
1	1	C?	.01/1206
2	1	C?	C1632C104J4RAC/SO8
3	1	C?	C1632C103J4RAC/SO8
4	1	C?	C1632C100J4GAC/SO8
5	1	C?	1uF/
6	16	C1,C2,C3,C4,C5,C6,C7,C8, C9,C10,C11,C12,C13,C14, C15,C16	.2/1206
7	14	C17,C393,C394,C395,C396, C397,C398,C399,C400,C401, C402,C403,C404,C405	0.1uF/1206
8	9	C18,C19,C20,C21,C22,C24, C25,C26,C27	.01 uF/
9	4	C23,C28,C29,C30	.01 uF/1206
10	18	C31,C32,C33,C34,C35,C36, C37,C38,C39,C41,C42,C45, C47,C50,C51,C52,C53,C54	.1/1206
11	18	R5,R12,R13,R14,R15,R16, R17,R18,R19,R20,R21,R22, C40,C43,C44,C46,C48,C49	100/1206

## Checking for Correct PCB Footprints

The next step after creating the BOM is to verify the following:

1. That every component in the design has a PCB Footprint attached
2. That the footprint referenced for every component is a legal and correct PCB Footprint

This isn't real easy. For instance, in the example above, item 8 (some capacitors) don't have a footprint attached. To fix this, each of these objects in the schematic must be found and the correct PCB Footprint property attached. Orcad does, thankfully, provide the **Export Properties** and **Import Properties** commands to ease this somewhat. Using **Export Properties**, one may dump all the properties of all components in the design to a text file. The file may then be edited to put in properties and the file reloaded into the design using **Import Properties**. Unfortunately, the file format is ugly and mistakes can be disastrous to the whole design.

A better method is to use Orcad's internal spreadsheet function. Open up a schematic sheet, then do **Edit...Find**. Leave the target as '\*', select **parts**, and then find. This should select all the parts but no wires, junctions, buses, or ports. Now hit **Ctrl-E** (or use **Edit...Properties**) and a *spreadsheet* of properties will magically appear:

	Reference	Designator	Value	PCB Footprint	Power Pins Visible	Primitive	Implementation Type
1	C31		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
2	C32		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
3	C33		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
4	C34		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
5	C35		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
6	C36		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
7	C37		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
8	C38		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
9	C39		.1	1206	<input type="checkbox"/>	DEFAULT	<none>
10	U13		CY7C13	PLCC52	<input type="checkbox"/>	DEFAULT	<none>
11	U11		CY7C00		<input type="checkbox"/>	DEFAULT	<none>
12	U12		CY7C00		<input type="checkbox"/>	DEFAULT	<none>

In this spreadsheet you may then edit the PCB Footprints more easily. This spreadsheet method is also a nice way to change a bunch of values. By the way, it works on any group of similar things. Try it on a group of nets or, when building a part in a library, on a bunch of pins. Columns and rows may be resized for viewing convenience.

Now, how do you know what legal footprints exist? Without a copy of Orcad Layout, that's a tough one. Fortunately, we have a copy available to us, and a file exists on the server for reference:

<D0server1.fnal.gov/projects/Trigger Electronics/CAE/PCB Footprints.txt>

To fix up the footprints in the schematic, go to every component and stuff in the correct value. For reference, in the example spreadsheet above, I should not use '1206' but instead should be using 'SM/C\_1206'.

## Associating Component Values to Procurement Data

Once a bill of materials is put together, it can then be imported into the Parts & Vendors program. Parts & Vendors uses the combined **Value/PCB Footprint** data from the Bill of Materials as the Part Number, which references a vendor and their catalog number. When the Bill of Materials is imported it creates a Parts List for the design, from which Parts & Vendors will create purchase orders – sorted by vendor, grouped by vendor, with all the right text and multiplied out for the number of boards being built. This data can then be printed out and, hopefully, just handed off to a secretary to enter in all your orders.

The astute reader will realize that Parts & Vendors also provides a database of stuff we've used before and can be used as a hint when building the schematic of the PCB Footprints that should be used. When the data is imported, P&V will match anything it can and then create new parts entries for anything that didn't match up. Those are the parts you'll have to look up the vendor data for. Everything else was done previously, so you don't have to find the Digi-Key part number for the millionth time.

To import the data to P&V, perform the following steps:

1. Use **File...Import Data From Text File...To New Parts List**.
2. In that dialog box, make the following selections:
  - 2.1. P/N from data
  - 2.2. Field separator TAB
  - 2.3. Field 1 is Item Number
  - 2.4. Field 2 is List Qty

2.5. Field 3 is Reference

2.6. Field 4 is Part Number

3. Then, browse to your file.
4. Create the new parts list name and hit 'Open File'.
5. For each line that has an item number, hit the Assign button. If you want, you can delete the lines of no interest, and then select everything and use Ctrl-A.

This will create the new parts list in P&V. Look it over and enter vendor data for any parts that didn't match up. Once everything is matched up, you may then use List Rollup to build a rollup of costs, see a breakdown of costs, and generate RFQs or P.O.s.